USFOS
Reality Engineering

USFOS AS

Phone: +47 905 05 717

www.USFOS.com

Enterprise No.: NO 986 827 374 MVA

MEMO

MEMO CONCERNS	FOR YOUR ATTENTION	COMMENTS ARE INVITED	FOR YOUR INFORMATION	AS AGREED
Release Notes **USFOS Version 8-7**				
DISTRIBUTION Members of USFOS user group				X

FILE CODE	CLASSIFICATION
	Confidential
REFERENCE NO.	

PROJECT NO.	DATE	PERSON RESPONSIBLE / AUTHOR	NUMBER OF PAGES
	2014-01-01	Tore Holmas	19

Release Notes

USFOS 8-7, Jan 2014

This memo contains project information and preliminary results as a basis for final report(s).
USFOS AS accepts no responsibility of this memo and no part of it may be copied.

1 INTRODUCTION 3

2 CHANGES IN VERSION 8-7 4

2.1 RESTRICTIONS ON SHELL ELEMENT ANGLES 4

3 NEWS IN USFOS VERSION 8-7 - 2014. 6

3.1 INTRODUCTION 6
3.2 HOW TO UPGRADE YOUR USFOS VERSION 6
3.3 ENHANCED GRAPHICAL USER INTERFACE 7
3.3.1 All-Time-High 7
3.3.2 Visualization of Local Axes 8
3.3.3 Visualization of Heavy Items 9
3.4 TUBULAR JOINT CAPACITY MODELS 10
3.4.1 NORSOK Revision 3 10
3.5 EARTHQUAKE 10
3.5.1 Delay 10
3.5.2 Stretching/Compressing 11
3.6 SUPPRESSING BUCKLING OF INTEGRATED PLATE-I-GIRDER 12
3.7 UNLOADING AFTER BOAT IMPACT 13
3.8 BUCKLING CURVES (CINIDEF) 13
3.9 LOAD PANELS FOR MAPPING TO FE-ELEMENTS 14
3.10 PUSHDOWN ANALYSIS (FIRE) 14
3.11 LOCAL AXIS OF ZERO-LENGTH SPRING (ELMTRANS) 14
3.12 SWITCHES, (SPECIAL OPTIONS). 15
3.13 UPDATES USFOS AND UTILITY TOOLS 16
3.14 NEW/MODIFIED INPUT COMMANDS 16
3.15 DOCUMENTATION 16

4 WARNINGS AND HINTS 18

4.1 JOINT MODELLING AND PILE-IN-LEG. WARNING. 18
4.2 USE OF NONSTRU AND LIN_ELEM. WARNING 18
4.3 BEAM HINGES. HINT. 19

1 Introduction

The current official version of USFOS is version 8-7 with release date 2014-01-01. The release contains the following:

- Release Notes (this MEMO)
- Updated software on www.usfos.com
- Extended examples library on www.usfos.com
- Updated manuals on www.usfos.com

Except for this MEMO, no written information will be distributed in connection with this release. All information is stored on the WEB.

2 Changes in version 8-7

In order to improve the quality of the analysis results, important changes are made:

2.1 Restrictions on shell element angles

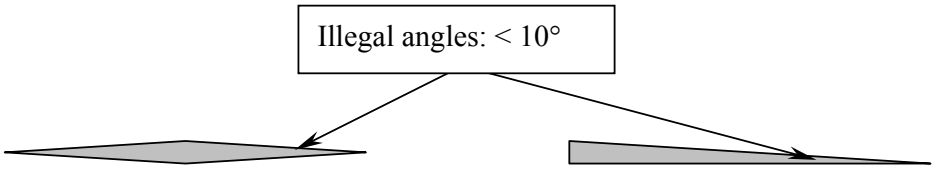


Figure 2-1 Illegal small angles for 4- and 3-node shell

Extreme **small** element corner angles are checked for. If the angle is less than 10°, USFOS will report this as an illegal angle and stop with an error message.

Following information are given:

- The elements are printed in the "out" file
- The USFOS label file: *"prefix"_illegal_shell_angles.usl*" could be opened in xact and the illegal elements are shown.

The best option is always to modify the structural model and remove the bad-shaped elements. However, if the user decides to keep the elements, the "**`ILLEGAL`**" command could be used to bypass the check.

Figure 2-2 describes the recommended "bypassing": Specification one-by-one. This option means that the user has an overview over important and less important elements, (which f ex remain elastic or have "no" practical load carrying function).

The option works as follows:

- A new minimum angle is defined by the user
- The element to accept are listed

The minimum angle could be re-defined several times. The commands are executed in the specified sequence.

```
'         Key-1      Opt      Value
Illegal ShellAngle  Accept   9      ! Redefine Min angle

'         Key-1      Opt   IDs
Illegal ShellAngle  Elem   1001  1002 ! Accept elem 1001 and 1002
                           2001  2002 ! Accept elem 2001 and 2002
                           3001  3002 ! Accept elem 3001 and 3002
```

Figure 2-2 Accepting elements with small angles. Specification element-by-element

The "lazy" (not recommended) version is to accept all short elements without any specification. A warning will be printed in the output file, (see Figure 2-4).

```
'         Key-1      Opt      Value
Illegal ShellAngle  Accept   5   ! Redefine Min angle

'         Key-1      Opt
Illegal ShellAngle   UsersRisk  ON   ! Accepting everything unchecked
```

Figure 2-3 Accepting elements with small angles without element specification

```
        ------------------------------------------------
        --    * * *    W A R N I N G   * * *           --
        --      Unconditionally Acceptance of         --
        --    Very Small Angles on Users own Risk     --
        ------------------------------------------------
```

Figure 2-4 The warning is printed in the output file.

In the output file (.out), the results from element checking are printed. If illegal elements are found, the analysis stops, and an USFOS Label file (name: *illegal_shell_angles.usl*) is created. By opening this file (File/Read Labels from file), a label with the angle is attached to each illegal shell in the graphical presentation of the model.

3 News in USFOS version 8-7 - 2014.

3.1 Introduction

The new features are described by examples in this memo and in the updated manuals. In addition to the usual platforms, (win32, LINUX-and maxosx-64), version 8-7 is also built on win64.
All utility software found under "modules" is built on 32bit platform and works therefore on both win32 and win64 computers.

3.2 How to upgrade your USFOS version

From release 8-6, USFOS could be upgrades in different ways:

- Alt 1: Download the new "*setup.exe*" and u-install/install USFOS, (same as for release 8-6). This operation requires administrator rights on the PC.
  This alternative is required if the 64 bit windows is installed.

- Alt 2: Download module by module and copy into the application folder, (typical "*C:\Program Files\USFOS\bin*". This operation requires write access on **C:**, but no administrator rights are required since no installation operations are performed, (just file copy).

With alternative 1, all modules and the on-line manuals are updated.

For alternative 2, (Win32 only), following should be done:

- Download USFOS module , unzip and copy into C:\Program Files\USFOS\bin
- Download xact, (complete package) , unzip and copy into C:\Program Files\USFOS\bin
- Download USFOS and xact user's manuals. Copy into C:\Program Files\USFOS\bin

Alternative 2 means that the existing files located on the Application folder will be over-written, (take a backup copy of the actual files if you want to keep your existing USFOS modules).

Similar procedure is used for other USFOS modules, (for example STRUMAN).

3.3 Enhanced Graphical User Interface

The graphical user interface (*xact*) has been enhanced since last year's release. The GUI version released together with USFOS 8-7 is "2.7" for the Win-64bit version. The functionality is the same on win32 and win64, but the win64 version has access to more memory.

3.3.1 All-Time-High

Element results are changing during an analysis. For example will the plastic utilization change during an earthquake simulation. This new option makes it possible to create one "all-time-high" presentation *up to the actual load level*. For example: If the last stored load step in the analysis is selected, the "all-time-high" for the entire analysis is displayed.

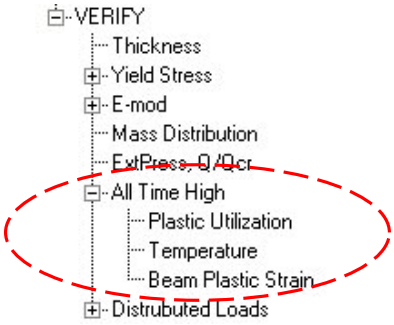


Figure 3-1 Verify All Time High menu

Figure 3-2 presents the usual "element plastic utilization" at point for full load (left) and for a later step (mid) after unloading of the force. The "all-time-high" (right) will remember the highest utilization of every member up to the actual load step.

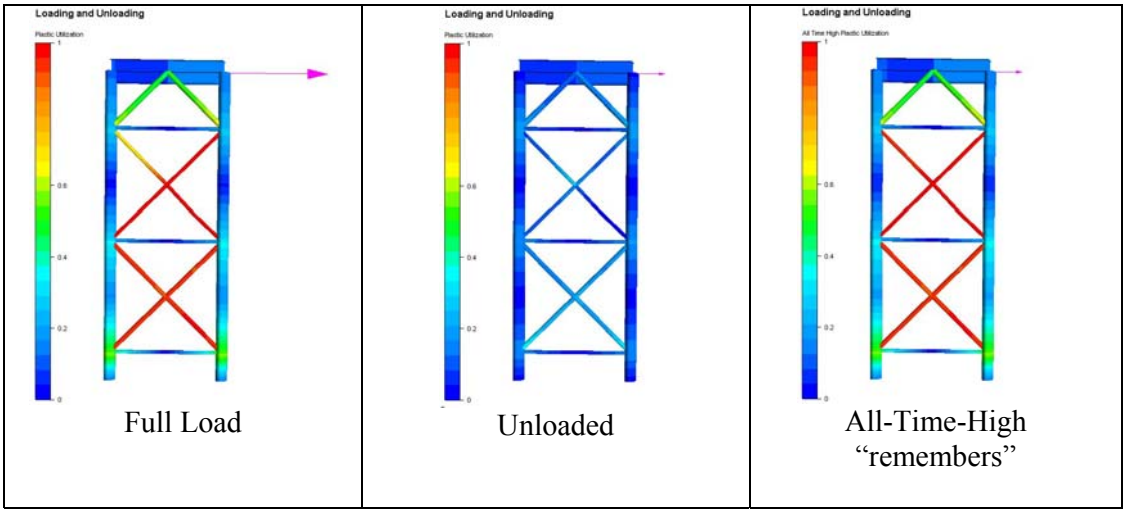


Figure 3-2 – "All-time-high" plastic utilization of cyclic loaded frame

3.3.2 Visualization of Local Axes

The local Z-axis of beam elements is visualized using “Global Vector/Verify/Local Z-Axis”. The arrow-length is automatically set relative to the model size, and the length could be changed using the “Verify/Vector Settings” menu. Clipping and display parts on/of could be used to zoom into actual element(s).

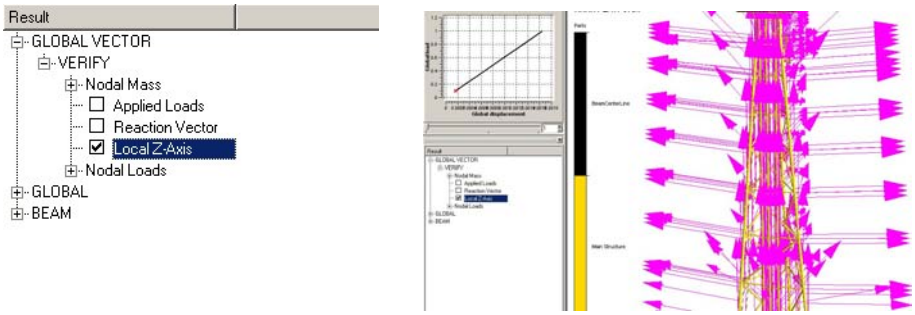


Figure 3-3 - Verification of Local Z-axis for Beam Elements

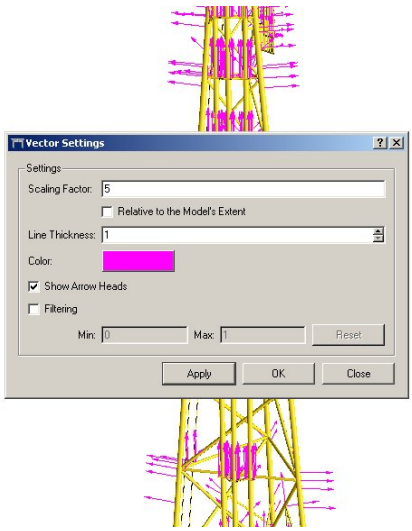


Figure 3-4 – Adjusting Arrow-size (Verify/Vector Settings)

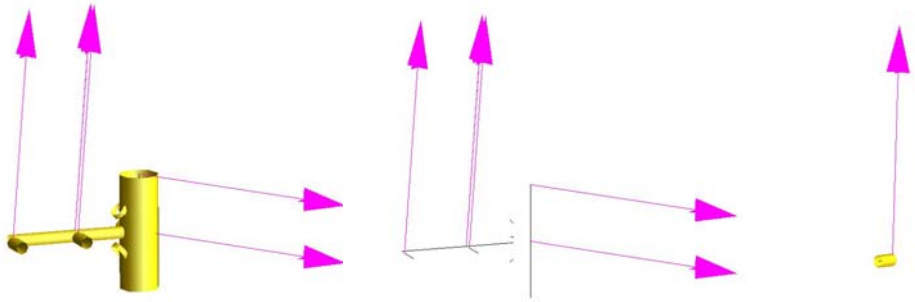


Figure 3-5 - Use Clip and show parts on/off to easier see actual elements

3.3.3 Visualization of Heavy Items

Heavy items are normally modelled using one or more NodeMass definitions. Node masses have been possible to visualize in xact for several years, (blue bars, and the lengths reflect the relative magnitude).

However, in order to improve the information from an analysis, some new equipment mass definitions are introduced (see user's manual). In the USFOS analysis, the masses from these 1- 2- and 4-node mass-elements are applied as usual node-masses, but in xact, the items are visualized as shown below.

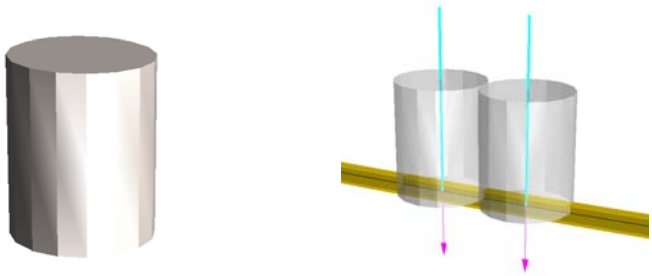


Figure 3-6 - 1-node equipment. Two items are shown to the right.

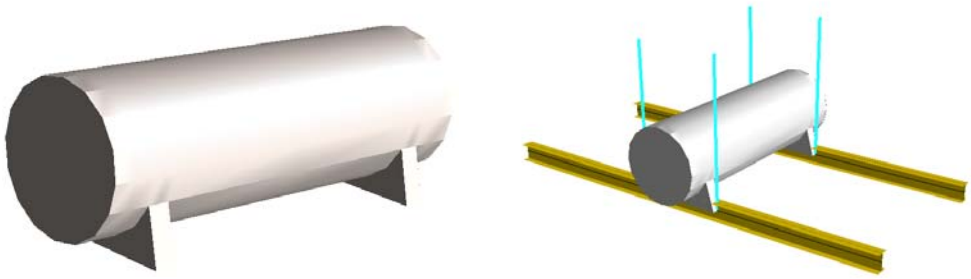


Figure 3-7 - 4-node equipment. The 25% of the total mass per node.

3.4 Tubular Joint Capacity Models

3.4.1 NORSOK Revision 3

A new capacity curve is added to the joint option: Norsok Revision-3. This option is activated with the following ChJoint command:

```
'         Node    Ch1   Ch2   Geo    Rule
 ChJoint  101      10    20    0     NOR_R3
```

Figure 3-8 Activating Norsok Revision-3 joint capacity

3.5 Earthquake

3.5.1 Delay

Often, ground motion time histories starts for time=0.0. If loads should be applied prior to the earthquake, and this takes some time, the earthquake could be delayed using the new option:

```
SWITCHES     Earthquake  Delay        ΔT
```

This means that *all* motion histories defined are delayed a certain time ΔT.

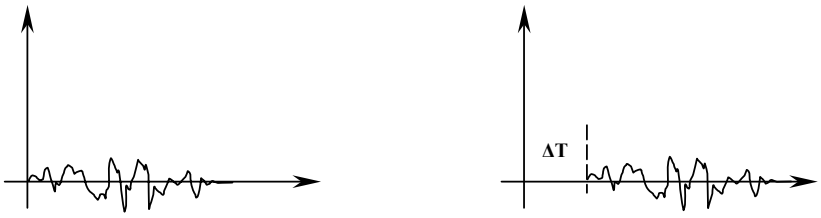


Figure 3-9 - Original Motion History (left) and delayed a time ΔT (right)

3.5.2 Stretching/Compressing

When a structure experiences an earthquake (ground motion), the response is highly dependent on the dynamics. If the ground moves with a period close to the fundamental eigen-periods of the structure, the response increases for every cycle.

Ground motions used in design are often based on recorded earthquakes adapted to the actual field.

In order to demonstrate the robustness of the structure, these design Time Histories could be "stretched" and "compressed" as shown in Figure 3-10. The stretching and compression means that the time axis is scaled with a factor >1 (stretching) or <1 (compressing).

By for example running the simulation with ±30% (stretching factor 1.3 and 0.7), it could be demonstrated that possible resonance periods are checked.

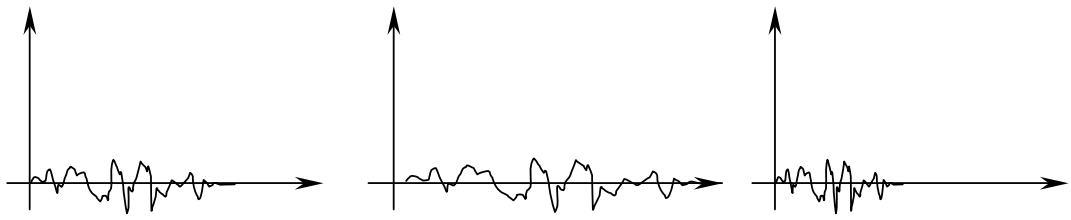


Figure 3-10 - Original Motion history (left), Stretched (mid) and Compressed (right)

By specifying the command:

```
SWITCHES    Earthquake  Stretch     Factor
```

All motion histories are stretched/compressed with the same factor.

3.6 Suppressing buckling of integrated plate-I-girder

The USFOS beam-column element predicts the correct buckling with only one element per physical member. When needed, a mid-node is inserted (internally) and the element is split into two ½ elements. This is normally an advantage since the user does not need to worry about mesh density to obtain the correct ultimate capacity.

However, in some cases this local member buckling is not wanted.

For example: If an I-girder is continuously welded to a plate, the girder is suppressed from buckling about the weak axis. Only buckling about the strong axis is possible.

By specifying:

```
SWITCHES    Solution    PlateEdge ON
```

USFOS will search for I-girders attached to plates and suppress buckling about the weak axis of the I-girder.

Figure 3-11 demonstrates an axially loaded panel, where the two I-girders buckle in a non-physical manner for a relatively low compression load. The image in the middle shows the behaviour with the weak-axis buckling is suppressed. The image to the right shows that the panel could buckle about the strong axis, (which the physical correct behaviour of this integrated structure).

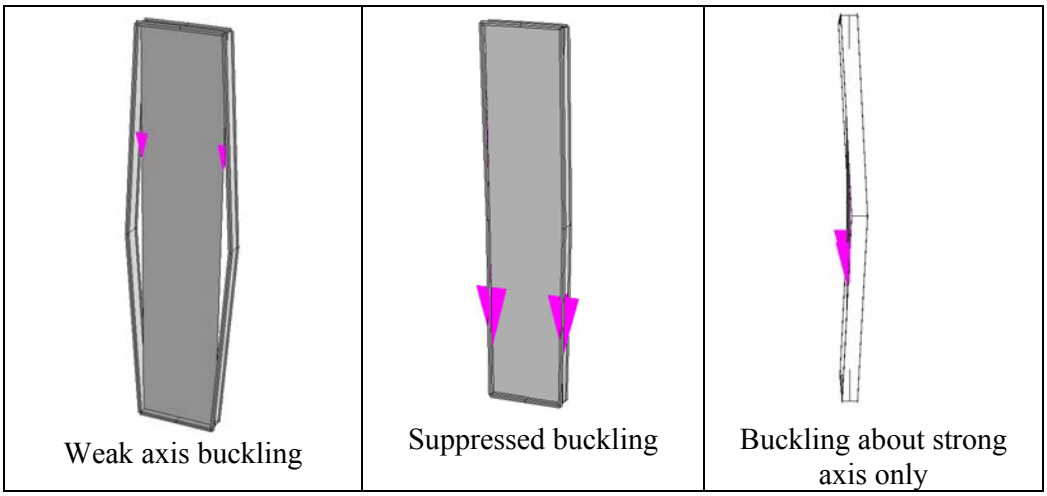


Figure 3-11 - Stiffened Plate. (shell + beam elements)

3.7 Unloading after boat impact

When a boat impact is completed (reached the energy level specified in the BIMPACT record), the impact force is unloaded to zero. Originally, the default increment during unloading was 0.1, but in version 8-7, this factor is reduced to 0.02 (i.e. 50 steps if no step scaling takes place).

In addition to this changed default, the user may control the load factor used during unload using the following command:

```
SWITCHES    Impact      UnLoFact    loadFactor

For example:
SWITCHES    Impact      UnLoFact    0.01  ! Unload 1% per step
```

3.8 Buckling Curves (CINIDEF)

Some of the buckling curves are changed slightly to become consistent with respect to the use of factors (load/ and material).

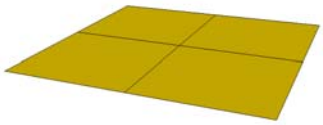
All buckling curves could be defined using the key-words given in the manual. The former "NPD" series are now renamed to EuroCode-3, (Euro3_A, Euro3_B, etc).

Cinidef introduces imperfection also on I- and Box sections. The strong axis parameters are used in the buckling formulas.

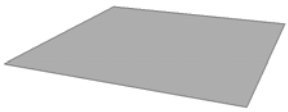
3.9 Load Panels for mapping to FE-elements

If for example an area load or explosion pressure should be mapped to a large number of shell elements, the new LOAFIELD option could be useful.

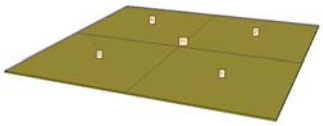
Shell elements located closer to the panel than a certain user-defined tolerance will get the actual pressure. The load orientation will follow the direction of the load panel, (unaffected about the local sequence numbering of each shell element).



4 shell elements



1 load panel



Shells and Panel

3.10 PushDown analysis (fire)

If fire degradation analysis should be performed, the PushDown command is recommended. This command will modify every element's section capacities based on the actual temperature field. This new option is an extension of the existing "FireChk" option, and is prepared for more detailed cross section temperature data from FAHTS.

3.11 Local Axis of Zero-length spring (ElmTrans)

In addition to the more general definitions of the "main element" for definition of the local coordinate system of a zero-length-spring, a direct definition is available. The keyword "MainBeam" is used to specify the element. See User's manual.

3.12 SWITCHES, (Special Options).

The command "SWITCHES" was introduced in 8-5 to switch on special options and is extended in version 8-7. Following "Switches" commands are available, (subkeys in bold are new):

KeyWord	SubKey	Value	Description	Default
General	*IndefLimit*		Min Max imperfection (in CINIDEF).	0.05 1 %
Defaults	*Version*	ver	850: switch to version 8-5 defaults	870
WaveData	*TimeInc*	val	Time between each hydrodyn calc.	Every
	NoDoppler	-	Switches OFF oppler effects.	ON
	NoStore		Switches OFF storing of wave data for visualiz.	ON
	TidalLevel	Level	Specify Tidal Level	0
	Accuracy	val	Change accuracy. 0: old accur, 1: new accur	1
	SeaDim	X , Y-dim	Specify size of sea surface used in xact	2 λ
NodeData	*DoublyDef*	ON/OFF	ON: Accept doubly defined nodes with same coo	OFF
StatusPrint	*MaxElem*	val	Max element in status print	10
Iterations	*RLF_Calc*	-	Activate "Residual Load Factor" method	OFF
Write	*FE_Model*	Case/stp	Writes deformed FE model at given case/stp	OFF
	LinDepAlt	-	Writes ZL-springs for each BLINDP2	Off
Solution	*FracRepeat*	MxRep	Max fracture repeates	10
	PlateEdge	ON/OFF	Avoiding I-girder to buckle about weak axis if the beam element is attached to a plate element	OFF
	Impact	*UnLoFact*	Load factor during unloading after boat impact	0.02
StrainCalc	*InclDent*	ON/OFF	OFF: not included. ON: included	ON
	Algorithm	val	0: old. 2: new, incremental.	2
	Visualization	ON/OFF	Including Gradients. ON/OFF	ON
Results	*ShellComp*	"val"	Number of shell results	5
WindData	*ReynDep*	ON/OFF	Switch to Reynold-number dependent Cd	OFF
EarthQuake	***Delay***	Val	Delays earthquake with specified time	0
	Stretch	Val	Stretches the motion history with specified value	1

Table 3-1 SWITCHES options

3.13 Updates Usfos and Utility Tools

News, corrections and updates are described on the web, and it is recommended to check the following link:

http://www.usfos.no/news/index.html

3.14 New/modified input commands

Since last main release (8-6), following input identifiers are added/extended:

PushDown	:	New command	: Fire Degradation analysis
LoaField	:	New command	: Load Field.
Equip_1N	:	New command	: 1-Node Mass Object.
Equip_2N	:	New command	: 2-Node Mass Object.
Equip_4N	:	New command	: 4-Node Mass Object.
CHJOINT	:	Extended command	*: Norsok Revision 3 joint curves.*
SWITCHES	:	Extended command	*: See above.*
ELMTRANS	:	Extended command	

3.15 Documentation

The following documentation, (updated or new), is available on the web:

- User's manual : Updated document
- PILERES users's manual : New
- JNTRES user's manual : New
- d2p user's manual : New

Tool	Description	Comment
d2p	Converts nodal displacements (nodeID + 6 displacement components) to prescribed, or forces displacement input to USFOS. (NODEDISP or NODELOAD+SPRNG2GR)	
JNTRES	Joint Strength overview. Extended Version	Needs version 8-7 or higher
PILERES	Total- and Pile reactions from static analysis	Needs version 8-7 or higher

Table 3-2 Misc Utility Tools

4 Warnings and Hints

4.1 Joint modelling and Pile-In-Leg. Warning.

When a pile goes inside a leg, the pile is forced to slide inside the leg. This is modelled in USFOS using the BLINDP2 command, where the pile node becomes a "slave" and the nearest leg elements becomes the "master element".

However, sometimes the user defines 2-node *springs* between the pile and the leg instead.

If the spring is attached to a leg node, which is used in connection with CHJOINT, problems will occur. The spring will become a "brace" and the load transfer through the joint will not work as intended.

4.2 Use of Nonstru and Lin_elem. Warning

Often, models created for linear elastic analysis are used directly for non-linear ultimate strength analysis. The quality of linear models varies much. Some models consist of components with sufficient strength to carry the functional loads, but too often, the models are unable to carry the self-weight. The reasons could be several: Early stage model (concept level), use of dummy elements and simply that the strength issue is pushed to the post-processing (code check) stage.

When such problems occur, it is tempting to "fix" the model using NONSTRU and LIN_ELEM commands.

Use of these commands in that context is another words for saying:

"My model is not suited for non linear analysis".

The NONSTRU command was introduced for member importance studies, where one or more elements were removed and the impact on the ultimate capacity was checked. Controlled use seldom causes problems, but defining whole areas non-structural, could cause concentrated load transfer into the remaining structure and then cause new problems.

The best solution is always to strip the model for surplus elements, but sometimes a careful change of material properties (increase yield and reduced stiffness) could be a good solution.

The LIN_ELEM command could be replaced by a re-definition of the material, for example scaling the yield with a certain factor. This gives a better control of the "non-physical" areas of the structure.

Material properties are easily changed using the CHG_MAT command, where the actual elements also could be referred to using the Group option.

4.3 Beam Hinges. Hint.

Beam hinges could cause numerical problems if a group of elements give zero stiffness.

If is possible to replace one hinge with a zero length spring + one extra node.

Struman has an option for automatic generation of springs, (the new Hing2Elm commands).

Contact support@USFOS.com for examples.